

Computational Fluid Dynamics Analysis Success Stories of X-plane Design to Flight Test

*Gary B. Cosentino
NASA Dryden Flight Research Center
Edwards, California*

NASA STI Program ... in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA scientific and technical information (STI) program plays a key part in helping NASA maintain this important role.

The NASA STI program is operated under the auspices of the Agency Chief Information Officer. It collects, organizes, provides for archiving, and disseminates NASA's STI. The NASA STI program provides access to the NASA Aeronautics and Space Database and its public interface, the NASA Technical Report Server, thus providing one of the largest collections of aeronautical and space science STI in the world. Results are published in both non-NASA channels and by NASA in the NASA STI Report Series, which includes the following report types:

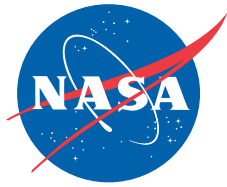
- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services also include creating custom thesauri, building customized databases, and organizing and publishing research results.

For more information about the NASA STI program, see the following:

- Access the NASA STI program home page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA STI Help Desk at (301) 621-0134
- Phone the NASA STI Help Desk at (301) 621-0390
- Write to:
NASA STI Help Desk
NASA Center for AeroSpace Information
7115 Standard Drive
Hanover, MD 21076-1320



Computational Fluid Dynamics Analysis Success Stories of X-plane Design to Flight Test

*Gary B. Cosentino
NASA Dryden Flight Research Center
Edwards, California*

National Aeronautics and
Space Administration

Dryden Flight Research Center
Edwards, California 93523-0273

NOTICE

Use of trade names or names of manufacturers in this document does not constitute an official endorsement of such products or manufacturers, either expressed or implied, by the National Aeronautics and Space Administration.

Available from:

NASA Center for AeroSpace Information
7115 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

ABSTRACT

Examples of the design and flight test of three true X-planes are described, particularly X-plane design techniques that relied heavily on computational fluid dynamics (CFD) analysis. Three examples are presented: the X-36 Tailless Fighter Agility Research Aircraft, the X-45A Unmanned Combat Air Vehicle, and the X-48B Blended Wing Body Demonstrator Aircraft. An overview is presented of the uses of CFD analysis, comparison and contrast with wind tunnel testing, and information derived from CFD analysis that directly related to successful flight test. Lessons learned on the proper and improper application of CFD analysis are presented. Highlights of the flight-test results of the three example X-planes are presented.

This report discusses developing an aircraft shape from early concept and three-dimensional modeling through CFD analysis, wind tunnel testing, further refined CFD analysis, and, finally, flight. An overview of the areas in which CFD analysis does and does not perform well during this process is presented. How wind tunnel testing complements, calibrates, and verifies CFD analysis is discussed. Lessons learned revealing circumstances under which CFD analysis results can be misleading are given. Strengths and weaknesses of the various flow solvers, including panel methods, Euler, and Navier-Stokes techniques, are discussed.

NOMENCLATURE

3D three-dimensional
CAD computer-aided design
CFD computational fluid dynamics
NASA National Aeronautics and Space Administration

BACKGROUND

During the 1980s the birth of the supercomputer and the enabled application of computational fluid dynamics (CFD) analysis techniques quickly advanced together in a dedicated development effort within the aircraft research and design industry. In particular, the efforts of the National Aeronautics and Space Administration (NASA) in cultivating the development of both the supercomputer and CFD were unsurpassed. At the NASA Ames Research Center (Moffet Field, California) an entire division of the organization was dedicated to obtaining and operating what was at that time the state-of-the-art supercomputer, with another division immersed in the development, application, and validation of CFD techniques. These early CFD algorithms were targeted toward and made great use of the newly procured supercomputers in a productive, interdependent relationship.

The gains made not only at NASA Ames but across the country during that decade were unprecedented, one technology complementing and enabling the other. The tremendous increases in computing speed and memory storage that were occurring almost on a quarterly basis enabled the ambitions and abilities of the researchers to grow at a matching rate. What took days or perhaps

weeks to compute (and therefore was too impractical to undertake) in the late 1970s could be completed in just hours with the advent of the Cray-1 computer (Cray Research Incorporated, Bloomington, Minnesota) by 1981.

Once this higher computing speed was possible, researchers could accordingly increase the scope of their computations, which yielded increasingly complex CFD analysis codes requiring the next generation of supercomputer, and so on. Both computing speed and calculation fidelity involving ever-increasing fluid physics representations grew rapidly throughout the decade.

The newly emerging computer architectures required new and unique ways of measuring speed and throughput. It was obvious that computing power was increasing rapidly, but evaluating that power quantitatively required the development of a set of standardized benchmarks that could be uniformly applied to the new supercomputers, and which tasked them in ways representative of the computational algorithms that were desired to be run in the day. The group at NASA Ames that was dedicated to obtaining and running the fastest computers available put together just such a set of benchmarks that measured, fairly and repeatedly, the power that was available to the researcher with each new generation of supercomputer operated at NASA Ames (ref. 1).

In the early 1980s CFD techniques were developed for calculation of flow fields about airfoils (two-dimensional) (2D) and wings (three-dimensional) (3D) using potential flow theory. Complex fluid physics including viscosity were neglected in this formulation, nonetheless, very useful aerodynamic solutions were obtained within reasonable amounts of central processing unit (CPU) time. With the advent of true supercomputers such as the Cray-1 in 1981, the CPU time required for some 3D potential solutions became so small (in some cases just 10 seconds), that a new possibility for CFD application emerged: performing computational design optimization. Toward that end, various researchers began looking at optimization algorithms and identified one method, the so-called quasi-Newton method, as a robust and general means of driving some objective function to a local (hopefully global) minimum. Combining this method with a fast CFD flow solver able to compute the lift-to-drag (L/D) ratio of a wing geometry in seconds or minutes of computer time rather than hours allowed the new supercomputers to not only analyze aerodynamics, but to alter the geometry to optimize aerodynamics, within some suitable constraints. One successful example of this achievement is given in reference 2.

As the 1980s progressed, the trend toward faster processors, multiple processors, and more memory (speed) continued. Periodic upgrades of computing power could be relied upon by the many researchers who were developing the complementary CFD algorithms for the new capabilities. The researchers were then ready to challenge the capacity of the latest installed supercomputer to cope with the new, more complex algorithms. Increasing fidelity of fluid physics were modeled by the new CFD codes, surpassing the relatively simple panel and potential equation solutions with the nonviscous Euler equations of motion (adding vorticity effects) and finally, the full viscous Navier-Stokes equations.

In parallel with the increasing fidelity of fluid physics came the increasing scope of geometry modeling. More and more dense 3D grids about complex geometries were created. Multiple body problems, internal flow problems, and even moving grids (store separation problems) were

undertaken. Computational times of hours became the status quo; it seemed as though the problems undertaken were matched to the supercomputer capacity available at the time that would result in turnaround times of a few to several hours. It was as though this was the “threshold of pain” of the researchers and engineers for the time they could wait to see their answers. Complex problems requiring days were generally avoided, if for no other reason than it could not be reasonably assured that the computer would remain “up” for that length of time. A crash, a reset, required maintenance, or a reboot would cause such a long-running problem to be lost. Thus, a balance was struck between the computing power available and the scope and ambition of the problem to be solved.

COMPUTATIONAL FLUID DYNAMICS ANALYSIS AND THE AIRCRAFT DESIGN PROCESS

When starting from a “clean sheet of paper,” the typical aircraft design process usually begins with a “configurator” laying out a rough sketch of the outer lines of the new shape on the computer-aided design (CAD) system at hand. After the overall planform and other shaping characteristics are decided upon, the designer will usually have enough information to create a 3D model of the aircraft: a model suitable for early, lower-order CFD analysis. Efficient, flexible CFD tools including panel and potential equation codes and other linear methods are available and are extremely useful at this early stage of design to generate preliminary aerodynamics for the new shape. This information allows redesign and refinement to proceed quickly, maturing the aircraft design almost in real time.

At this point in the design process there is sufficient detail in the CAD system to begin discretizing the shape and preparing the electronic model of the aircraft for more detailed and refined analysis, including for both CFD and finite-element structural codes to be brought into the process. It is generally at this point that some state-of-the-art CFD methods are applied and detailed fluid dynamics generated. Based on these results, further design refinement can occur, going back to the CAD model upon which the original analysis was based. This iterative process can continue until the designers agree that the new aircraft shape meets initial criteria and performance characteristics, and is worthy of still more complex analysis, to include building a model of the aircraft for wind tunnel testing.

At this point in the design process CFD analysis plays a crucial, if not its most important, role. Wind tunnel models are generally very expensive to build, costing perhaps hundreds of thousands of dollars or more, and wind tunnel test time is a significant cost driver during a project. Viscous CFD analysis methods applied to the candidate geometry before cutting metal for the wind tunnel model generally comprise time and effort that is extremely well spent. Such early analysis has made the difference between building a costly, disappointing model and one that simply verifies the adequacy of the design as predicted by the CFD analysis methods. A “no surprises” wind tunnel is generally the goal at this stage of the design process. As part of the wind tunnel test and CFD analysis stage, CFD analysis can provide a link from the model configuration and data to the actual aircraft configuration.

To support the model in the test section of the wind tunnel, a modification is generally made at the aft end of the model to allow the wind tunnel sting to be inserted into the aft end of the model. The forward end of the sting is connected to the strain gage balance inside the model for force and moment resolution, and the aft end of the sting is affixed to the wind tunnel test section. The modification of the shape at the aft end of the model to accommodate the typically cylindrical sting is called the sting distortion. The data taken is for the shape of the model with this distortion, not the shape of the actual aircraft (the undistorted shape). Computational fluid dynamics analysis can provide a unique and generally quite accurate sting distortion correction, allowing the designers to correlate, using this correction, the forces and moments measured in the wind tunnel to what the forces and moments would be on the actual, undistorted aircraft shape. One way to accomplish this correction is to model the aircraft shape in CFD with a perfectly expanded jet plume shape emanating from what would be the engine nozzle; this method should provide the forces and moments on the actual aircraft shape as they would be in flight with the engine plume. The results can then be correlated to the wind tunnel measurements, which represent a distorted body shape and a cylindrical solid plume (the sting). As a double-check, the designer also can model the sting distortion shape in CFD using all of the increments for correlation and prediction of actual aircraft performance.

Wind tunnel models typically have several pressure orifices in order to measure the pressure distribution on the surface of the model. In contrast, CFD analysis solutions provide surface pressures on all locations on the surface of the model, limited only by the grid density (discretization) of the surface. Good correlations, therefore, can be made between the surface pressure measurements at the few locations on the physical model, and the overall pressure distribution predicted by CFD analysis of the model. If the correlations are good, the designer can be relatively confident that the forces and moments predicted by the CFD analysis are good as well. If the correlations are not good, the wind tunnel data can be used to calibrate or even improve the CFD analysis method by making improvements to the calculation method or grid density, and having the solutions rerun.

The relationship of CFD analysis and wind tunnel testing is synergistic and complementary. Experience verifies the benefits of utilizing CFD analysis early in the aircraft design phase. Early CFD analysis can assist the designer (and configurator) in shaping the aircraft in a preliminary way to meet early performance criteria and can greatly aid in designing a wind tunnel model that can be reasonably expected to perform well. Gross errors in design are usually predicted well enough by CFD analysis to allow corrections to be made with confidence in the model design.

When a wind tunnel model design has been agreed upon and cutting the metal for the model begins, CFD analysis can continue before the model enters the wind tunnel, in order to fill out the database of flow conditions for later reference once the wind tunnel model data begins to be generated. It can be extremely useful to have the CFD-generated aerodynamic database available during the wind tunnel test, to correlate immediately with the data coming out of the test (in real time). In one instance, the wind tunnel testing was stopped because the wind tunnel test data did not correlate at all with the CFD analysis predictions, and in fact an error in the wind tunnel data reduction parameters was discovered and corrected. Without the correlation capability, this wind tunnel testing and data acquisition would have continued with the data reduction parameter error unnoticed, requiring a complete recalculation of the wind tunnel database after the test if and when

the error was discovered.

COMPARING AND CONTRASTING COMPUTATIONAL FLUID DYNAMICS ANALYSIS WITH WIND TUNNEL TEST DATA

Using CFD analysis effectively in the manner described in the previous section, and upon detailed comparisons of the CFD analysis predictions to the wind tunnel data, several conclusions are generally evident. Assuming that a robust and accurate Navier-Stokes flow solver (and sufficiently dense flow field grid) was used, typical strengths and weaknesses of the CFD analysis are revealed. For flow conditions of low angle of attack or sideslip (benign flow with little or no separation), the CFD-analysis-predicted forces and moments, as well as surface pressure distributions, are generally found to be in good to very good correlation with the wind tunnel test data. Once a few conditions of this nature are checked, the CFD analysis method can be used with good confidence for examining the flow field in detail, perhaps with an eye toward minor redesign of the shape. This can help tremendously in progressing the design closer to a prototype aircraft post-wind-tunnel test. Because the CFD analysis was found to be in good agreement with the wind tunnel test data at these benign conditions, it could be concluded that it might be safe to trust the method to aid in refining the model shape further, without, perhaps, building a new wind tunnel model and retesting. This saves an expensive and time-consuming step for the next iteration of the design process.

Another strength of CFD analysis for benign flow conditions is extraction of the increment from the results of the sting distortion mentioned earlier. The overall predicted L/D ratio of the design will be altered in the wind tunnel test data due to the presence of the sting and the attendant distortion of the aft section of the aircraft model. Computational fluid dynamics analysis of the undistorted shape will produce the increment of this alteration in performance. Thus, the wind tunnel test data, with CFD-generated corrections, can be recalculated to better represent the aircraft design in flight. This is routinely performed for aircraft configurations, especially those in which the sting is inserted at the aft end (the engine nozzle end) of the aircraft model, and is an extremely useful correction technique.

The success with which CFD analysis can be compared to wind tunnel test data is usually confined to the benign flow conditions of small to moderate angle of attack. For higher transonic Mach numbers, where strong shocks are prominent on the aircraft shape, the ability of CFD analysis to cope with and accurately predict flow field separation, shock-induced separation, or massive flow separation at very high angles of attack is limited at best. Highly viscous-dominated flow conditions, wherein many simplifying assumptions used in formulating CFD analysis codes are not valid, create the limitations evidenced upon comparing both the forces and moments and the surface pressures to the wind tunnel test data. What is generally found is that, for example, the lift curve slope of the aircraft configuration is followed quite well over the small to moderate angle of attack range, but above this range, near the onset of lift breakdown, the slope generated by the CFD analysis methods will diverge from the wind tunnel test data. The CFD analysis methods can still give an approximation of where the lift curve “knee” will appear, but likely will not be very

accurate in the calculation of maximum lift coefficient (CL_{max}), for example. It may also be found that upon examining the surface pressure distributions, the shock location, strength, and sharpness will be inaccurate. The stronger the shock waves, generally the poorer will be the CFD analysis calculations of their strength. Specifically, over the range of approximately Mach numbers 0.90 to 1.10, the strength of the shocks and their ability to separate the boundary layer are difficult for CFD analysis methods to model accurately. Increase the Mach numbers to, for instance, 1.2 and above, however, and the calculations again become more accurate in terms of forces and moments and surface pressure distribution predictions.

Drag calculation is another general area of CFD analysis weakness. Since drag is, for most aircraft configurations, small when compared to the lift and moment forces, inaccuracies play a larger role in the values obtained. Also, since drag onset due to separation is largely a viscous-dominated flow characteristic, the extent of separation is difficult for CFD analysis to compute well. This weakness manifests itself when comparing drag polars of the configuration to the wind tunnel test data. It should be mentioned that even for the benign flow conditions and lower angles of attack, the absolute drag computed by the CFD analysis method may be “off” by an almost constant increment over the entire range of the test data, diverging finally at the more severe conditions. This increment may come about from the difference between the calculated skin friction drag and the wind tunnel test data. The increment can sometimes be determined to be fairly constant at a given Mach number, therefore, upon examination of the data, it may be possible to “correct” this incremental difference by adding a constant to the CFD-analysis-calculated drag by way of post-processing the computed data and then replotting the CFD analysis results with the wind tunnel test data superimposed, effectively compensating for the incremental error in the absolute drag numbers.

Other major areas of comparison include the moment curves and the derivation of stability derivatives. Often, again for the more moderate flow conditions, these curves are fairly linear, and CFD analysis can do a very good job of predicting these quantities. Asymmetrical calculations of a model at sideslip conditions in CFD analysis require a full grid, without taking advantage of a symmetry plane. Therefore, these calculations double the time and computing resources required. Once comparisons with wind tunnel data are made and found to be favorable, investing these resources to compute asymmetric stabilities derivatives becomes worthwhile.

Surface pressure distributions are easily compared (if the wind tunnel model is so equipped) and for flow conditions where good comparisons exist, the CFD-analysis-derived pressures can then be used for other detailed analysis of the configuration. For example, the dense surface pressure data can be used in conjunction with a finite-element structural model for calculation of loads and moments about the aircraft.

The all-important calibration of the sting distortion increment is usefully provided by CFD analysis methods, allowing extrapolation of the characteristics of the wind tunnel model (with sting distortion) to the actual aircraft configuration with the correct aft shaping. This correction makes the wind tunnel test data even more useful for prediction of full-scale aircraft performance parameters.

Again, one of the best uses of CFD analysis is to ensure that the wind tunnel model that is built yields a largely “no surprises” wind tunnel test. Once the CFD analysis methods are so calibrated from one test, it is possible to apply them with even greater confidence to the next design iteration, even perhaps allowing refinements to be made to the design without the need for a subsequent wind tunnel reentry. This is clearly an area in which an investment in CFD analysis is worthwhile.

DIFFICULT AREAS FOR APPLICATION OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS

The previous section discussed “benign or moderate” flow conditions. These conditions are typically the low to medium angles of attack or sideslip, and the low transonic or low supersonic Mach numbers. Once significant flow separation is present, or at high transonic Mach numbers (approximately 0.90 to 1.10) where very strong shocks are present, discrepancies with test data are likely to be prominent. In typical CFD analysis codes used for full aircraft configuration analysis, turbulence is generally modeled to some approximation in order to provide a reasonably sized problem. The various turbulence models do a fairly good job for areas of no to small separated flow. Once the separation becomes significant, with large areas of stagnated and recirculating flow, these models generally break down. The result is the under- or overprediction of the separated regions, with the attendant inaccuracies in the surface pressure distribution and integrated forces and moments. Where very strong shocks are present, first the shock strength and location are usually poorly predicted, and then the resulting flow separation and recirculation regions are accordingly mispredicted. When applying CFD analysis under these conditions, great caution should be used unless there are test data to either validate the results or to calibrate the errors of the computations.

Even under benign flow conditions, CFD analysis can be misleading when applied to certain regions of the flow field of the aircraft shape. For example, applying CFD analysis in a boattail region, perhaps in an aft-facing step area or in the area of the exhaust nozzle, significant flow separation can exist even for benign flow conditions. Drag calculations for a configuration with aft-facing steps will likely be inaccurate. Configurations with landing gear in the flow stream are similarly troublesome. Landing gear are often complex shapes, both difficult to model in the computational grid, and difficult to compute for the CFD flow solver. It is often desired to evaluate the increment of drag with landing gear down versus landing gear retracted, resulting in the temptation to use CFD analysis methods to evaluate this early in the design stage. Caution should be exercised in these areas of interest unless wind tunnel test data is available to calibrate and correct the results.

APPLICATION OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS TO INTERNAL FLOWS

Most of the discussion thus far has been made in the context of CFD analysis of the external shape of an aircraft configuration for the purposes of force and moment calculation and surface pressure distribution, but CFD analysis can be applied very effectively to the calculation of internal

flows, specifically the calculation of inlet and nozzle flows including the ducting before and aft of a simulated engine. In fact, many more detailed wind tunnel models have provision for so-called “flow-through” configurations, wherein an inlet is uncovered and flow is allowed to enter the inlet, flow through the model, and exit at the aft end of the model near the sting area. To control the flow through this ducting, different inserts can be fabricated to choke down the flow at the exit, thus giving the effects of varying mass flow.

To correctly compute the flow about and through such a model, it is necessary to grid and model the internal flow, including the effects of varying mass flow. This modeling allows calculation and prediction of inlet drag and inlet spill. These effects appear clearly in the surface pressures near and immediately behind the inlet, affecting the drag and base pressures near the sting. Including these effects in the CFD analysis model can greatly improve the agreement of the computations with the wind tunnel test data. In fact, if the CFD analysis is seriously used in the detailed design of the wind tunnel model, these effects should be included.

In addition to flow-through models, purely propulsion-related internal flows can also be effectively computed. Inlet and ducting designs can be fairly accurately assessed provided the attendant flow separations are reasonably subdued. Reference 3 outlines some detailed internal flow calculations and their comparisons with experimental data. Similarly, nozzle flow paths can be designed using internal CFD analysis computations with care toward and knowledge of the inherent limitations. These are generally more complex flow problems than purely external calculations, yet very good results can be obtained that aid greatly the designer’s efforts to refine a configuration before committing it to costly metal fabrication and testing. In addition, CFD analysis can help determine the best location and placement of wind tunnel model surface instrumentation (for example, pressure taps).

APPLICABILITY OF COMPUTATIONAL FLUID DYNAMICS ANALYSIS RESULTS TO SUCCESSFUL FLIGHT TEST

Applying CFD analysis to aircraft design in the manner described above, despite its limitations, has been instrumental in allowing a well-performing wind tunnel model to be built and enabling the extrapolation of both computed and measured quantities to full-scale flight-test articles. Many of the corrections to wind tunnel test data, extrapolation of wind tunnel test data, and modeling of physical features of an actual aircraft play a major role in ensuring flight-test success. Often, the X-plane aircraft design will undergo many design changes after the wind tunnel model and the testing process are completed. The changes may be minor, but some quantification of the effect on aerodynamics or performance must be made in order to develop the flight control software. Computational fluid dynamics analysis, especially after having been compared (and somewhat calibrated) to the wind tunnel test data, can be confidently used in assessing these changes to the flight article, and the increments used to modify the database for inclusion in flight control law development. This was done repeatedly during the design and testing process for each of the three X-plane examples to be given in this report, and the flight-test results reflect the benefits of such an approach. In short, the CFD analysis corrections, when properly applied where they are valid, can be more accurate than simple extrapolation. In particular, stability derivatives can be computed

from the CFD analysis forces and moments, and over the regions of validity (benign to moderate flow conditions) have been found to be quite accurate during flight test. For neutrally to highly unstable aircraft configurations, accurate calculation of these stability derivatives is critical.

THE X-36 TAILLESS FIGHTER AGILITY RESEARCH AIRCRAFT

The design process of the X-36 Tailless Fighter Agility Research Aircraft (McDonnell Douglas, now the Boeing Company, Chicago, Illinois) proceeded almost exactly in the same manner as the process described in the above sections of this report. The X-36, shown in figs. 1 and 2, proved, both by way of full-scale simulation and subscale flight testing, that it is possible to achieve fighter-class agility in a configuration without vertical control surfaces. Yaw control on the X-36 was provided entirely by split ailerons (drag rudders) and thrust vectoring.



EC97-44294-2

Figure 1. The X-36 Tailless Fighter Agility Research Aircraft in flight over the Mojave Desert.



EC97-44294-8

Figure 2. The X-36 Tailless Fighter Agility Research Aircraft in flight.

Extensive analysis was performed on both this and similar precursor configurations using the full spectrum of computational analysis tools available to the designer. Each of the various types of CFD analysis methods was used at the appropriate time during configuration development, and each refinement of the design was aided significantly by the CFD analysis that was performed. As the design matured, more sophisticated CFD analysis methods were employed.

Critical to this process was an assessment of the stability and control derivatives of the design, because instability was a given byproduct of the goals of the aircraft. Thus, candidate configurations were assessed using the various analytical methods that were applied in determining the levels of instability, confirming that they were tractable given the capabilities of the modern digital flight control systems of the day. Occasionally, the analysis tools showed unacceptable levels of instability, and design adjustments were made to bring the configuration back within the acceptable limits of stability and control.

Before beginning the design of the X-36 wind tunnel model, extensive full Navier-Stokes CFD analysis was performed on the wind tunnel test configuration (including the sting distortion), providing valuable data to fine-tune the model shape and to assist in determining the location of instrumentation on the model. Once the model was committed to metal, further extensive CFD analysis runs were performed, pre-running many of the cases planned to be run during the wind tunnel test, and producing a database similar to what was expected to be generated during the wind tunnel test. When the actual data acquisition began during the wind tunnel test, the CFD database was in place and operational for early comparisons with the wind tunnel test data, available almost in real time as the data was provided by the wind tunnel test data system. It was this capability that enabled early detection and correction of an error in the data reduction scheme in the wind tunnel test data calibrations.

As a result of the preliminary analysis using full Navier-Stokes CFD analysis, the initial results of the wind tunnel test data indicated, as desired, “no surprises:” the model designed was performing exactly as had been hoped and as had been indicated from the CFD analysis database. Further, the piloted simulation of the aircraft using the stability derivatives of the configuration as analyzed by CFD was not only entirely flyable by the test pilot, but also met the maneuverability and agility goals that were defined early in the program.

From the successes described above, the decision was made to take the next step, which was to build a subscale flyable prototype that would be remotely piloted from a fixed ground station and hand flown by the same pilots who evaluated the handling characteristics in the simulations performed earlier using the combination of CFD analysis and wind-tunnel-derived stability coefficients. The success of this flight test was exemplary; the X-36 flew 33 safe and successful research flights from 1997 to 1998. An assessment of the success of the flight test and stability and control characteristics is found in reference 4.

THE X-45A UNMANNED COMBAT AIR VEHICLE

The X-45A Unmanned Combat Air Vehicle (UCAV) (The Boeing Company, Chicago, Illinois) followed in the footsteps of the X-36 in that it too was configured without vertical control surfaces. In the case of the X-45A, shown in figs. 3 and 4, thrust vectoring and oppositely deflected outboard elevons in a so-called “crow mix” were used for yaw stability and control. The design evolution of the X-45A followed a very similar, if abbreviated, path of CFD analysis, design refinement, wind tunnel model design, and flight article design and fabrication. The success of the X-36 design approach to tailless, highly unstable configuration design and control led to a tremendously successful X-45A flight-test program. Sixty-four safe and successful research flights were conducted on two X-45A demonstrators, some of which were dual vehicle flights. References 5 and 6 provide a much more extensive description of the X-45A UCAV program and flight tests.



ED02-0295-5

Figure 3. The X-45A Unmanned Combat Air Vehicle in flight over the Mojave Desert.



EC05-0129-02

Figure 4. The X-45A Unmanned Combat Air Vehicle in flight.

THE X-48B BLENDED WING BODY DEMONSTRATOR

The configuration of the X-48B Blended Wing Body (BWB) research aircraft (The Boeing Company, Chicago, Illinois) has been of interest and under investigation for over 20 years. Extensive CFD analysis encompassing all methods described in this report have been utilized to establish and refine the aerodynamics of this unique configuration. Until very recently, no actual flight-test data had been obtained. The X-48B BWB, shown in figures 5 through 7, represents the first attempt to obtain quality, scalable data from both wind tunnel model tests and subscale flight tests.



ED06-0198-62

Figure 5. The X-48B Blended Wing Body research aircraft at rest on the Rogers Dry Lake bed.



ED07-0164-1

Figure 6. The first flight of the X-48B Blended Wing Body research aircraft over the NASA Dryden (Edwards Air Force Base) flight-test complex.



ED07-0164-04

Figure 7. The X-48B Blended Wing Body research aircraft on approach to its first landing.

The Boeing Company has subcontracted to Cranfield Aerospace (Cranfield, Bedford, United Kingdom) the design and fabrication of two high fidelity 8.5 percent scale models of a notional prototype (full-scale) aircraft. Careful attention has been paid to the scalability of data in order to accurately infer the flight characteristics of the full-size aircraft from the subscale flight test. The design of the shape of this aircraft has been developed and refined over many years using both CFD analysis and limited wind tunnel test data.

Upon completion of the first of the two Cranfield Aerospace-built models, a wind tunnel entry at the NASA Langley Research Center (Hampton, Virginia) Full-Scale Wind Tunnel was performed in March of 2006. The vehicle used was an identical match to the second, flightworthy model, which would undergo subsequent flight test. An extremely high-quality wind tunnel database for the X-48B BWB was obtained in over five consecutive weeks of testing. This database, with CFD analysis supplements, was used to produce the flight control law derivations that would later be used to control and stabilize the aircraft in flight.

Again, years of careful and methodical design, using the best analytical and CFD analysis tools, and calibrations with wind tunnel data, yielded an excellent flying aircraft. The X-48B BWB first flew on July 20, 2007, and has since flown more than 15 successful research flights. The pilot's comment that the aircraft flies very much like the simulator validates the quality of the analysis tools and that the work of the design and flight-test team was worthwhile and accurate. No surprises were encountered on the first 15 flights, and the handling qualities of the aircraft as reported by the pilot during the various flight-test maneuvers have been consistently stated to be excellent.

The X-48B BWB will be flown into late 2008. The expected outcome is high-quality flight-test data fully characterizing the aerodynamic and stability and control derivatives of the configuration sufficient to significantly reduce the risk associated with taking the next step of designing and building a much larger, manned prototype aircraft.

SUMMARY

This report presented and described examples of successful application of computational fluid dynamics (CFD) analysis to the design process of three true X-planes. The process of conceptual design, computer-aided design modeling and refinement followed by CFD analysis methods application and further refinement was described. Specifically, how CFD analysis can aid in the design of a wind tunnel model that will yield few if any surprises during wind tunnel testing was explained. In the wind tunnel, data can be directly correlated to the computed CFD analysis database, thus calibrating the CFD analysis methodology and in some cases validating the wind tunnel data reduction method.

Computational fluid dynamics analysis can be and has been an enabling technology in the development process of getting a new aircraft shape into flight. Controlling an inherently unstable configuration is critically dependent on determination of the aerodynamics and stability derivatives of the configuration; CFD can provide preliminary estimates of these quantities accurately enough for the development of early control laws and a flyable simulation. Configuration assessments and incremental redesign can then be accomplished in a deliberate fashion, with the goal of arriving at a final configuration committed to a more detailed analysis, leading toward a flight model with greatly improved chances of success.

REFERENCES

1. Bailey, D. H., et al., "The NAS Parallel Benchmarks – Summary and Preliminary Results," *Proceedings of the 1991 ACM/IEEE Conference on Supercomputing*, ACM, New York, 1991, pp. 158-165.
2. Cosentino, G. B., and T. L. Holst, *Numerical Optimization Design of Advanced Transonic Wing Configurations*, NASA-TM-85950, May 1984.
3. Won, Mark J., Michael D. Wong, Michael J. Fletcher, and Gary B. Cosentino, "Application of Various Flowsolvers to a Subsonic Serpentine Diffuser Geometry," AIAA 96-2648, July 1996.
4. Balough, Dwight L., "Determination of X-36 Stability Margins Using Real-Time Frequency Response Techniques," AIAA-98-4154, August 1998.
5. Cosentino, Gary B., and Michael J. Hirschberg, "Flight Testing Transformational Air Power: The Joint Unmanned Combat Air System (J-UCAS) X-45A Demonstrator," *ITEA Journal of Test and Evaluation*, Vol. 25, No. 4, pp 1-14.
6. Wise, Kevin A., "X-45 Program Overview and Flight Test Status," AIAA 2003-6645, September, 2003.

REPORT DOCUMENTATION PAGE					<i>Form Approved OMB No. 0704-0188</i>	
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.</p> <p>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</p>						
1. REPORT DATE (DD-MM-YYYY) 01-05-2008		2. REPORT TYPE Technical Memorandum			3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE Computational Fluid Dynamics Analysis Success Stories of X-plane Design to Flight Test				5a. CONTRACT NUMBER		
				5b. GRANT NUMBER		
				5c. PROGRAM ELEMENT NUMBER		
6. AUTHOR(S) Cosentino, Gary B.				5d. PROJECT NUMBER		
				5e. TASK NUMBER		
				5f. WORK UNIT NUMBER		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Dryden Flight Research Center P.O. Box 273 Edwards, California 93523-0273					8. PERFORMING ORGANIZATION REPORT NUMBER <div style="text-align: center;">H-2838</div>	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001					10. SPONSORING/MONITOR'S ACRONYM(S) <div style="text-align: center;">NASA</div>	
					11. SPONSORING/MONITORING REPORT NUMBER <div style="text-align: center;">NASA/TM-2008-214636</div>	
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified -- Unlimited Subject Category 02 <div style="float: right; text-align: right;"> Availability: NASA CASI (301) 621-0390 Distribution: Standard </div>						
13. SUPPLEMENTARY NOTES Cosentino, NASA Dryden Flight Research Center. An electronic version can be found at http://dtrs.dfrc.nasa.gov or http://ntrs.nasa.gov/search.jsp . Also presented at the 2007 ITEA Symposium, Kauai, Hawaii, November 12–15, 2007.						
14. ABSTRACT <p>Examples of the design and flight test of three true X-planes are described, particularly X-plane design techniques that relied heavily on computational fluid dynamics (CFD) analysis. Three examples are presented: the X-36 Tailless Fighter Agility Research Aircraft, the X-45A Unmanned Combat Air Vehicle, and the X-48B Blended Wing Body Demonstrator Aircraft. An overview is presented of the uses of CFD analysis, comparison and contrast with wind tunnel testing, and information derived from CFD analysis that directly related to successful flight test. Lessons learned on the proper and improper application of CFD analysis are presented. Highlights of the flight-test results of the three example X-planes are presented. This report discusses developing an aircraft shape from early concept and three-dimensional modeling through CFD analysis, wind tunnel testing, further refined CFD analysis, and, finally, flight. An overview of the areas in which CFD analysis does and does not perform well during this process is presented. How wind tunnel testing complements, calibrates, and verifies CFD analysis is discussed. Lessons learned revealing circumstances under which CFD analysis results can be misleading are given. Strengths and weaknesses of the various flow solvers, including panel methods, Euler, and Navier-Stokes techniques, are discussed.</p>						
15. SUBJECT TERMS Aircraft design, Computational fluid dynamics (CFD), Flight test, Wind tunnel correlation, X-planes						
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT <div style="text-align: center;">UU</div>	18. NUMBER OF PAGES <div style="text-align: center;">21</div>	19b. NAME OF RESPONSIBLE PERSON STI Help Desk (email: help@sti.nasa.gov)	
a. REPORT U	b. ABSTRACT U	c. THIS PAGE U			19b. TELEPHONE NUMBER (Include area code) <div style="text-align: center;">(301) 621-0390</div>	